

Vibrating Membrane

Introduction

In the following example you compute the natural frequencies of a pre-tensioned membrane using the 3D Membrane interface. This is an example of “stress stiffening”; where the transverse stiffness of a membrane is directly proportional to the tensile force.

The results are compared with the analytical solution.

Model Definition

The model consists of a circular membrane, supported along its outer edge.

GEOMETRY

- Membrane radius, $R = 0.25$ m
- Membrane thickness $h = 0.2$ mm

MATERIAL

- Young’s modulus, $E = 200$ GPa
- Poisson’s ratio, $\nu = 0.33$
- Mass density, $\rho = 7850$ kg/m³

CONSTRAINTS

The outer edge of the membrane is supported in the transverse direction. Two points have constraints in the in-plane direction in order to avoid rigid body motions.

LOAD

The membrane is pre-tensioned by in the radial direction with $\sigma_i = 100$ MPa, giving a membrane force $T_0 = 20$ kN/m.

Results and Discussion

The analytical solution for the natural frequencies of the vibrating membrane given in [Ref. 1](#) is:

$$f_{ij} = \frac{k_{ij}}{2\pi R} \sqrt{\frac{T_0}{h\rho}} \quad (1)$$

The values k_{ij} are derived from the roots of the Bessel functions of the first kind.

In [Table 1](#) the computed results are compared with the results from [Equation 1](#). The agreement is very good. The mode shapes for the first six modes are shown in [Figure 1](#) through [Figure 6](#). Note that some of the modes have duplicate eigenvalues, which is a common property for structures with symmetries.

TABLE 1: COMPARISON BETWEEN ANALYTICAL AND COMPUTED NATURAL FREQUENCIES

Mode number	Factor	Analytical frequency (Hz)	COMSOL result (Hz)
1	$k_{10} = 2.4048$	172.8	172.8
2	$k_{11} = 3.8317$	275.3	275.3
3	$k_{11} = 3.8317$	275.3	275.3
4	$k_{12} = 5.1356$	369.0	369.1
5	$k_{12} = 5.1356$	369.0	369.1
6	$k_{20} = 5.5201$	396.6	396.7

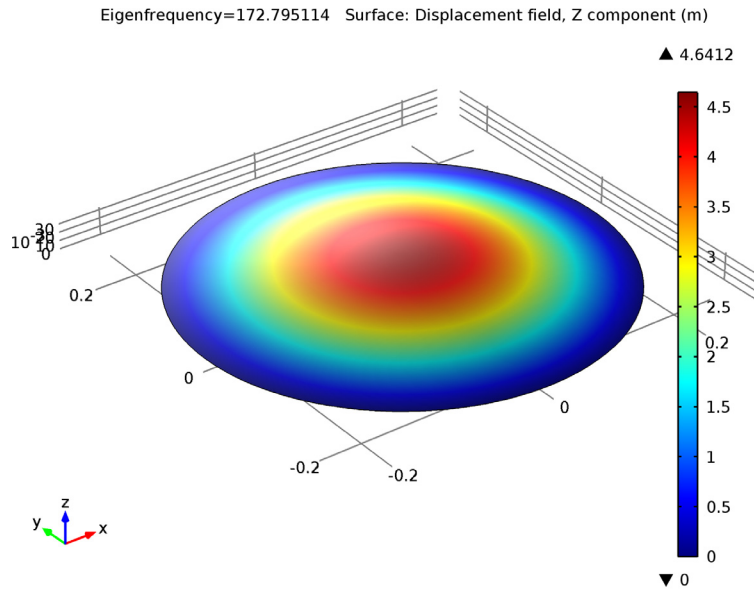


Figure 1: First eigenmode.

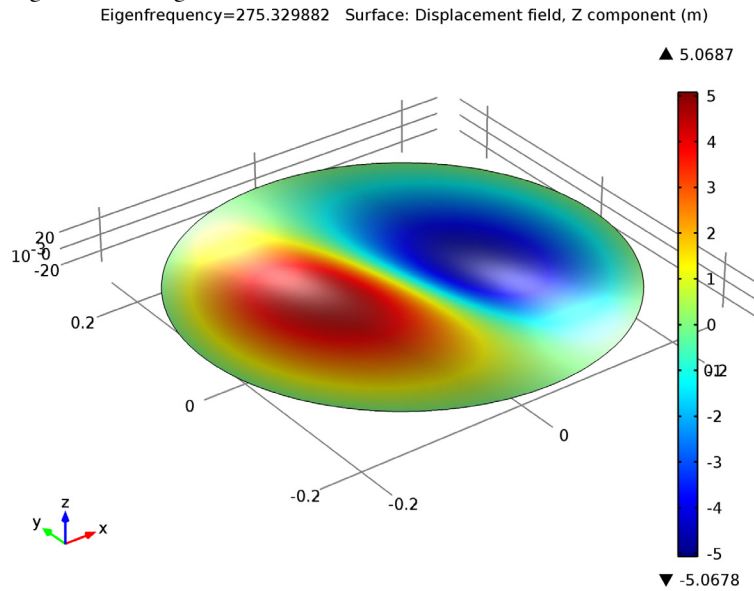


Figure 2: Second eigenmode.

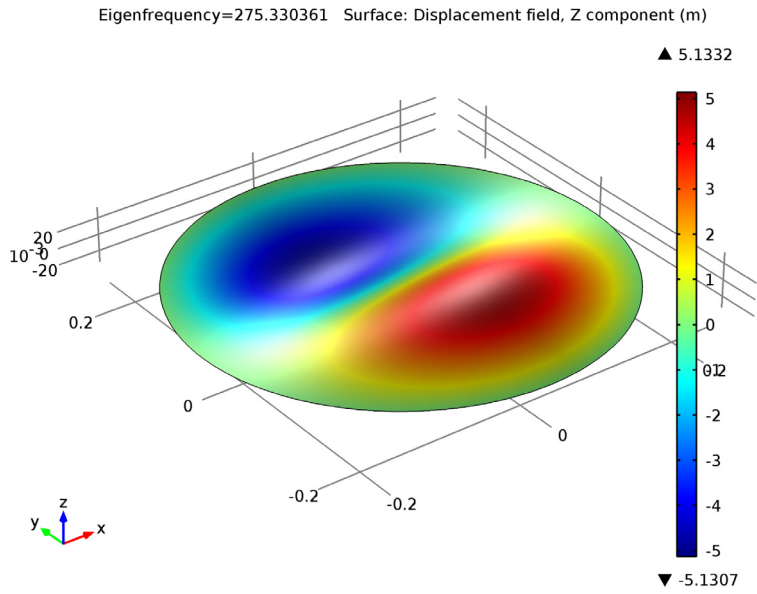


Figure 3: Third eigenmode.

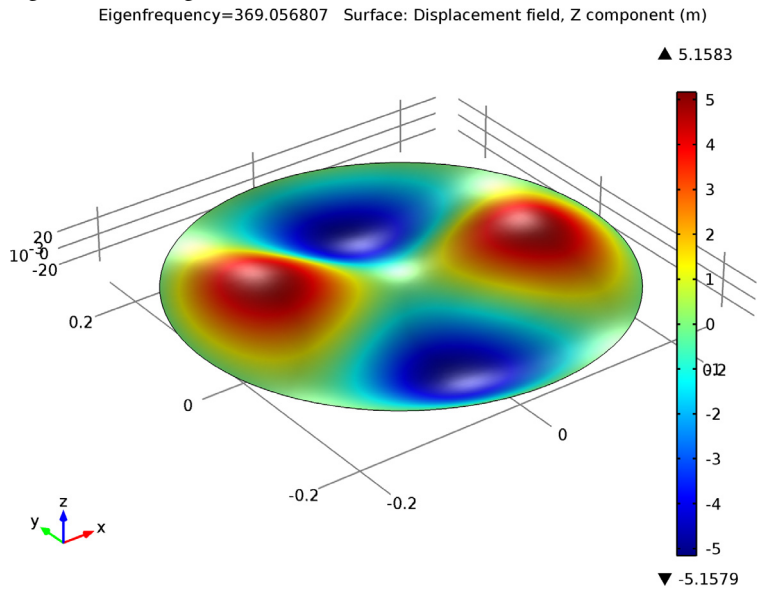


Figure 4: Fourth eigenmode.

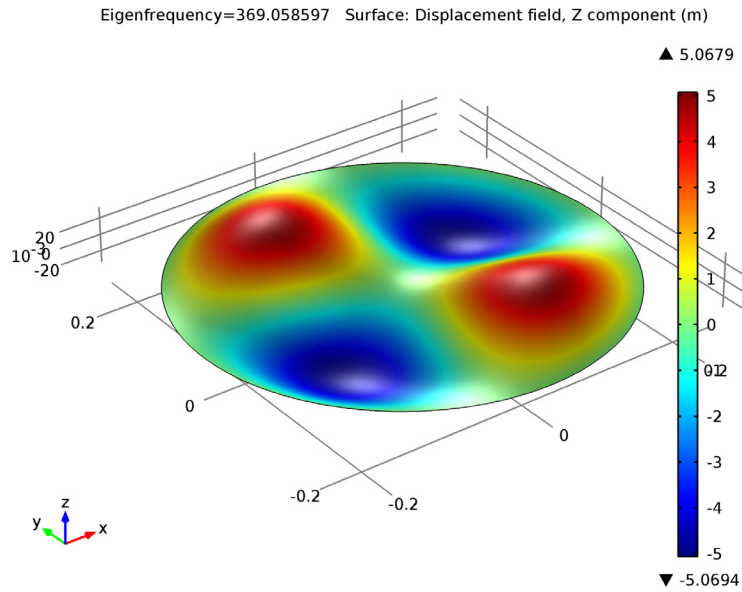


Figure 5: Fifth eigenmode.

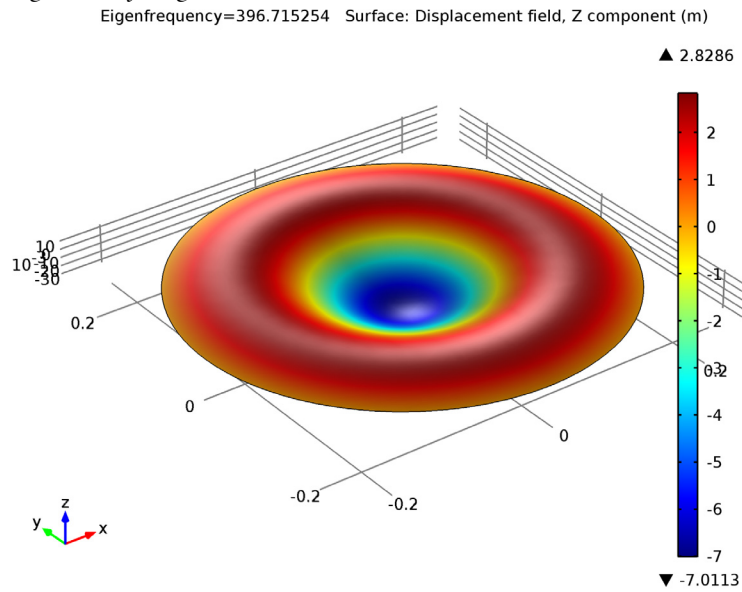


Figure 6: Sixth eigenmode.

Notes About the COMSOL Implementation

An eigenfrequency simulation with a pre-stressed structure can be simulated in two ways. If stresses are known in advance, it is possible to use an initial stress condition. This is shown in the first study.

In a general case, the prestress is given by some external loading, and is thus the result of a previous step in the solution. Such a study would consist of two steps: One stationary step for computing the prestressed state, and one step for the eigenfrequency. The special study type **Prestressed Analysis, Eigenfrequency** can be used to set up such a sequence. This is shown in the second study in this example.

Since an unstressed membrane has no stiffness in the transverse direction, it will generally be difficult to get an analysis to converge without taking special measures. One such method is shown in the second study: A spring foundation is added during initial loading, and is then removed.

Reference

1. A. Bower, *Applied Mechanics of Solids*, CRC Press, 2010.

Model Library path: Structural_Mechanics_Module/Verification_Models/
vibrating_membrane

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

- 1** In the **Model Wizard** window, click the **3D** button.
- 2** In the **Select Physics** tree, select **Structural Mechanics>Membrane (mem)**.
- 3** Click the **Add** button.
- 4** Click the **Study** button.
- 5** In the tree, select **Preset Studies>Eigenfrequency**.

6 Click the **Done** button.

GLOBAL DEFINITIONS

Parameters

- 1 On the **Home** toolbar, click **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
R	250[mm]	0.2500 m	Radius
thic	0.2[mm]	2.000E-4 m	Thickness
T0	100[MPa]*thic	2.000E4 N/m	Pre-tension force
E1	200[GPa]	2.000E11 Pa	Young's modulus
rho1	7850[kg/m^3]	7850 kg/m ³	Density
nu1	0.33	0.3300	Poisson's ratio
fct	$\sqrt{T0 / (thic * rho1)} / (2 * pi * R)$	71.85 1/s	Common factor in natural frequencies
f10	2.4048*fct	172.8 1/s	1st natural frequency
f11	3.8317*fct	275.3 1/s	2nd and 3d natural frequencies
f12	5.1356*fct	369.0 1/s	4th and 5th natural frequencies
f20	5.5201*fct	396.6 1/s	6th natural frequency

DEFINITIONS

On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

GEOMETRY I

On the **Geometry** toolbar, click **Work Plane**.

Circle 1 (c1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** right-click **Plane Geometry** and choose **Circle**.
- 2 In the **Circle** settings window, locate the **Size and Shape** section.
- 3 In the **Radius** edit field, type R.

MATERIALS*Material 1 (mat1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **New Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E1	Pa	Basic
Poisson's ratio	nu	nu1	l	Basic
Density	rho	rho1	kg/m ³	Basic

MEMBRANE (MEM)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Membrane (mem)**.
- 2 In the **Membrane** settings window, locate the **Thickness** section.
- 3 In the d edit field, type `thic`.

Initial Stress and Strain 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Membrane (mem)** right-click **Linear Elastic Material 1** and choose **Initial Stress and Strain**.
- 2 In the **Initial Stress and Strain** settings window, locate the **Initial Stress and Strain** section.
- 3 In the N_0 table, enter the following settings:

T0	0
0	T0

Prescribed Displacement 1

- 1 On the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.
- 2 Select all four edges.
- 3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in z direction** check box.

Fixed Constraint 1

- 1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.
- 2 Select Point 1 only.

Prescribed Displacement 2

- 1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
- 2 Select Point 2 only.
- 3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in y direction** check box.

MESH I

- 1 In the **Model Builder** window, under **Component I (comp1)** click **Mesh I**.
- 2 In the **Mesh** settings window, locate the **Mesh Settings** section.
- 3 From the **Element size** list, choose **Fine**.

STUDY I

On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (mem)

- 1 In the **Model Builder** window, expand the **Mode Shape (mem)** node, then click **Surface I**.
- 2 In the **Surface** settings window, locate the **Expression** section.
- 3 In the **Expression** edit field, type w .
- 4 On the **Mode Shape (mem)** toolbar, click **Plot**.
- 5 In the **Model Builder** window, click **Mode Shape (mem)**.
- 6 In the **3D Plot Group** settings window, locate the **Data** section.
- 7 From the **Eigenfrequency** list, choose the first frequency at **275.3** Hz.
- 8 On the **Mode Shape (mem)** toolbar, click **Plot**.
- 9 From the **Eigenfrequency** list, choose the second frequency at **275.3** Hz.
- 10 On the **Mode Shape (mem)** toolbar, click **Plot**.
- 11 From the **Eigenfrequency** list, choose the first frequency at **369.1** Hz.
- 12 On the **Mode Shape (mem)** toolbar, click **Plot**.
- 13 From the **Eigenfrequency** list, choose the second frequency at **369.1** Hz.
- 14 On the **Mode Shape (mem)** toolbar, click **Plot**.
- 15 From the **Eigenfrequency** list, choose **396.715254**.

16 On the **Mode Shape (mem)** toolbar, click **Plot**.

Now, prepare a second study where the prestress is instead computed from an external load.

ROOT

On the **Home** toolbar, click **Add Study**.

ADD STUDY

- 1** Go to the **Add Study** window.
- 2** Find the **Studies** subsection. In the tree, select **Preset Studies>Prestressed Analysis, Eigenfrequency**.
- 3** In the **Add Study** window, click **Add Study**.

MEMBRANE (MEM)

Edge Load 1

- 1** On the **Physics** toolbar, click **Edges** and choose **Edge Load**.
- 2** Select all four edges.
- 3** In the **Edge Load** settings window, locate the **Coordinate System Selection** section.
- 4** From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.
- 5** Locate the **Force** section. Specify the \mathbf{F}_L vector as

T0	r
0	phi
0	a

Add a spring with an arbitrary small stiffness in order to suppress the out-of-plane singularity of the unstressed membrane.

Spring Foundation 1

- 1** On the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.
- 2** Select Boundary 1 only.
- 3** In the **Spring Foundation** settings window, locate the **Spring** section.
- 4** Specify the \mathbf{k}_A vector as

0	x
---	---

0	y
10	z

Switch off the initial stress, which should not be part of the second study. In the eigenfrequency step, the stabilizing spring support must also be removed.

STUDY 2

Step 1: Stationary

- 1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.
- 2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mem)>Linear Elastic Material 1>Initial Stress and Strain 1**.
- 5 Click **Disable**.

Step 2: Eigenfrequency

- 1 In the **Model Builder** window, under **Study 2** click **Step 2: Eigenfrequency**.
- 2 In the **Eigenfrequency** settings window, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mem)>Linear Elastic Material 1>Initial Stress and Strain 1** and **Component 1 (comp1)>Membrane (mem)>Spring Foundation 1**.
- 5 Click **Disable**.
- 6 On the **Home** toolbar, click **Compute**.

RESULTS

Mode Shape (mem) 1

The eigenfrequencies computed using this more general approach are the same as before, except some small numerical differences.

To make **Study 1** behave as when it was first created, the features added for **Study 2** must be disabled.

STUDY I

Step 1: Eigenfrequency

- 1 In the **Eigenfrequency** settings window, locate the **Physics and Variables Selection** section.
- 2 Select the **Modify physics tree and variables for study step** check box.
- 3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mem)>Edge Load 1** and **Component 1 (comp1)>Membrane (mem)>Spring Foundation 1**.
- 4 Click **Disable**.